INTRODUCTION

EnSight supports files as written from ABAQUS (a commercial FEM solver) with certain restrictions. Versions 5.2, 5.3, and 5.6 are supported. Restrictions:

1. The geometry file (.fil suffix) must be ASCII. To obtain an ASCII geometry file, use the ABAQUS command `*FILE FORMAT,ASCII`. ASCII files are used for portability. However, reading large files can be slow.
2. Any desired element-based results must be nodal averaged, otherwise they are ignored. To obtain nodal results, use the ABAQUS command `*EL FILE,POSITION=AVERAGED AT NODES`.
3. For versions prior to 5.6, if a .dat file with the same root name as the .fil file is present in the same directory and contains element set information, EnSight will build parts based on those sets. Otherwise, all elements are combined into a single part. To place element sets in the .dat file, use the ABAQUS command `*PRE-PRINT,MODEL=YES`. (For version 5.6, element set data is contained in the .fil file and is always used.)

Reading data into EnSight is a two-step process. First, the appropriate files are selected. This step is largely the same regardless of the format of the data being read (see **How To Read Data** for more information). Second, parts are constructed. For ABAQUS data, parts are built automatically: a single part if no .dat file exists or parts based on the element sets in a .dat file if one is found.

ABAQUS datasets consist of the following files. Note that the entry in the File Name column is only a suggestion – it typically doe not matter to EnSight what the actual file name is (except that a .dat file must have the same root name as the .fil file).

File	File Name	Notes	Required?
Geometry	file.fil	Contains coordinates, element connectivity, and variables	required
Element sets	file.dat	Provides element set information so EnSight can create parts (for versions < 5.6)	optional

SEE ALSO

How To Read Data

User Manual: **Reading ABAQUS Data Files**